N8127093

ENERGY EFFICIENT ENGINE, HIGH-PRESSURE TURBINE COOLING MODEL TECHNOLOGY REPORT

PRATT AND WHITNEY AIRCRAFT GROUP
EAST HARTFORD, CT

MAY 1981

N8127093

1. Report No. NASA-CR-165374	2. Government Accession No.
4. Title and Subtitle: Energy Efficient Engine, High-Pressure Turbine Cooling Model Technology Report	5. Report Date: May 1981
	6. Performing Organization Code
7. Author(s): W. B. Gardner	8. Performing Organization Report No. PWA-5594-159
	10. Work Unit No.
9. Performing Organization Name and Address: United Technologies Corporation, Pratt & Whitney Aircraft Group - CPD, East Hartford, Connecticut 06108	11. Contract or Grant No.
	13. Type of Report and Period Covered: Contractor Report
12. Sponsoring Agency Name and Address: National Aeronautics and Space Administration, Washington, D.C. 20546	14. Sponsoring Agency Code

15. Supplementary Notes

Prepared in cooperation with NASA Project Manager C. C. Ciepluch
NASA Lewis Research Center, Cleveland, Ohio

16. Abstract

The objective of the High-Pressure Turbine Cooling Model Supporting Technology Program was to verify, through flow visualization tests, the acceptability of the high turbine blade cooling passage design for the Energy Efficient Engine.

Two-dimensional flow visualization model tests substantiated the flow stability benefits derived from the use of turning vanes in the root and tip turn flow areas and also indicated the need for corner fillets and flow injection into the acute corner formed by the intersection of the rib and simulated airfoil suction surface in order to minimize recirculation (stagnation) of flow in that region. Three-dimensional flow visualization model tests verified the actual blade coolant passage design.

17. Key Words (Suggested by Author(s)): Energy Efficient Engine; Cooling Flow Injection; Blade Cooling Design	18. Distribution Statement

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price*
Unclassified	Unclassified		

* For sale by the National Technical Information Service, Springfield, Virginia 22161

NASA-C-168 (Rev. 10-75)

FOREWORD

The Energy Efficient Engine Component Development and Integration Program is being conducted under parallel National Aeronautics and Space Administration contracts to Pratt & Whitney Aircraft Group and General Electric Company. The overall project is under the direction of Mr. C. C. Ciepluch. Mr. John W. Schaefer is the NASA Assistant Project Manager for the Pratt & Whitney Aircraft effort under NASA Contract NAS3-20646, and Mr. Michael Vanco is the NASA Project Engineer responsible for the portion of the project described in this report. Mr. William B. Gardner is Manager of the Energy Efficient Engine Project at Pratt & Whitney Aircraft Group.

TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

LIST OF ILLUSTRATIONS (Continued)

1.0 SUMMARY

The cooling model test program was one of several supporting technology programs structured to provide guidance to the Energy Efficient Engine high-pressure turbine component design effort. It accomplished its task by verifying, through water flow visualization test, the acceptability of the cooling flow passage designs. These designs enable the blade structural and life requirements to be met while at the same time permitting a blade design that can be readily fabricated.

Two-dimensional flow visualization model tests substantiated the flow stability benefits derived from the use of turning vanes in the root and tip turn flow areas and also indicated the need for corner fillets and flow injection into the acute corner formed by the intersection of the rib and simulated airfoil suction surface in order to minimize recirculation (stagnation) of flow in that region. Some tailoring of the root turn flow area was also required.

Three-dimensional flow visualization model tests verified the actual blade coolant passage design following the addition of five pedestals in the trailing edge tip turn area to eliminate a flow separation problem there.

2.0 INTRODUCTION

The objective of the National Aeronautics and Space Administration (NASA) Energy Efficient Engine Development and Integration program is to develop, evaluate, and demonstrate the technology for achieving lower installed fuel consumption and lower operating costs in future commercial turbofan engines. NASA has set minimum goals of 12 percent reduction in thrust specific fuel consumption, 5 percent reduction in direct operating cost, and 50 percent reduction in performance degradation for the Energy Efficient Engine (flight engine) relative to the JT9D-7A reference engine. In addition, environmental goals on emissions (the proposed EPA 1981 regulation) and noise (1978 FAR 36 standards) have been established.

In a high performance gas turbine engine utilizing cooled turbine blades, the objective is to maximize cooling effectiveness with a minimum of cooling airflow because excessive cooling flow can be detrimental to the achievement of engine performance goals. Internal cooling passage geometry must therefore be defined which will effectively distribute available cooling flow to achieve the desired blade metal temperatures while permitting a blade design that meets structural and life requirements and can be readily fabricated.

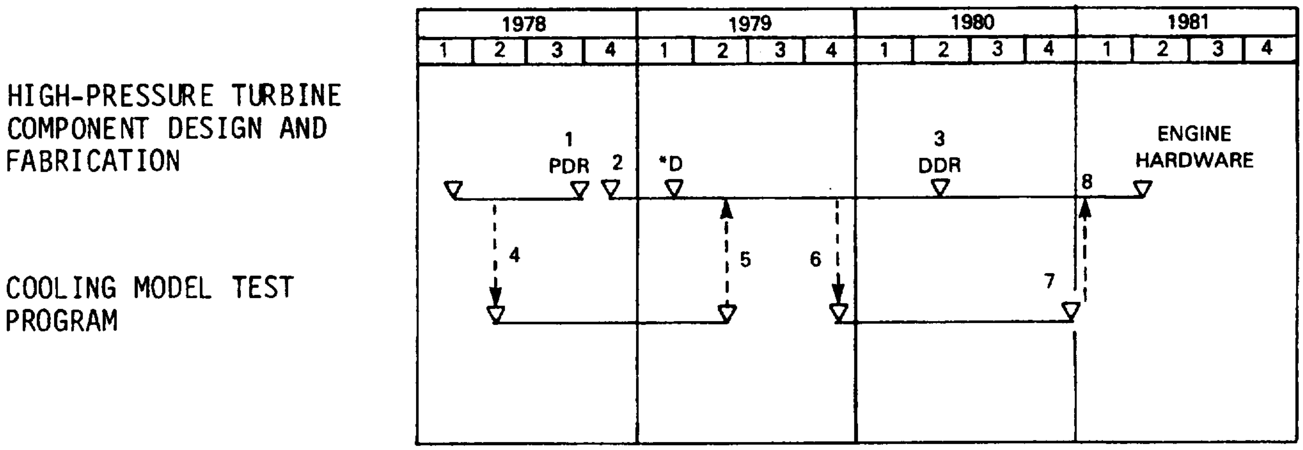
The purpose of the Turbine Cooling Model Supporting Technology Program was to verify, by water flow visualization tests, that the internal cooling flow passage design for the high-pressure turbine component blade did provide the desired flow distribution.

Plexiglas flow visualization test rigs five times actual blade size were designed and fabricated to simulate the blade internal geometry. A two-phase test effort was formulated. In the first phase, two-dimensional models were utilized to evaluate flow conditions in the critical root- and tip-turn passage regions. The second phase utilized a three-dimensional model which more accurately simulated the total cooling passage design.

Following initial model testing, regions of flow recirculation or stagnation were evaluated, design modifications incorporated, and retesting conducted to verify that the desired flow distribution was subsequently achieved.

This supporting technolgy program was conducted to permit timely interaction with the high-pressure turbine component effort, thereby ensuring that the evaluated flow control concepts could be readily incorporated into the component blade design (see Figure 2-1).

This document covers the definition of the blade cooling geometry, the analysis, design, and fabrication of test rigs to simulate this geometry, and test results obtained.



*D - Critical Decision Point for Selection of Vane and Blade Aerodynamics.
1 - Component Preliminary Design Completed.
2 - Component Detailed Design Initiated.
3 - Component Detailed Design Completed.

4 - Blade Internal Cooling Passage Geometry Provided to Test Program from Component Preliminary Design Effort.
5 - Two-Dimensional Model Test Results Define Modifications Required in Root and Tip Flow Turning Passages.
6 - Modified Blade Internal Cooling Passage Geometry Provided to Test Program for Three-Dimensional Model Definition.
7 - Internal Cooling Flow Geometry for Component Blade Verified.
8 - Design Modified

Figure 2-1 Cooling Model Program Logic Diagram.

3.0 ANALYSIS AND DESIGN

Analytical modeling of the complex flow characteristics within the cooling passages of advanced design cooled turbine blades is a time-consuming and expensive process. Experience has shown that flow visualization, through the use of low-cost simulation models, can provide the desired results in a much more efficient manner. The purpose of the analysis and design effort was to design flow visualization models that would accurately simulate the internal cooling flow geometry of the energy efficient engine high-pressure turbine blade.

3.1 COMPONENT DESIGN BASIS

The high pressure turbine component blade preliminary design formed the basis for the design of the initial flow models. The salient characteristics of the internal cooling flow configuration are illustrated in Figure 3-1. This blade design is characterized by severe twist and taper from root to tip, thin walls, and a narrow trailing edge wedge angle. The leading edge is cooled both by internal convection and by showerhead external film. The blade surface is cooled by air passing through three spanwise passages separated by high height-to-thickness ratio ribs. The flow exits through pedestals at the trailing edge. Trip strips are included in the passages to enhance convective heat transfer in those passages. Turning vanes are included in the root and tip turn areas of the multipass cooling system to reduce pressure losses in the turns and suppress formation of recirculation zones along the wall surfaces of the ribs. The blade tip is cooled both by internal convection and external film cooling. The desired coolant flow distribution for this configuration is illustrated in Figure 3-2.

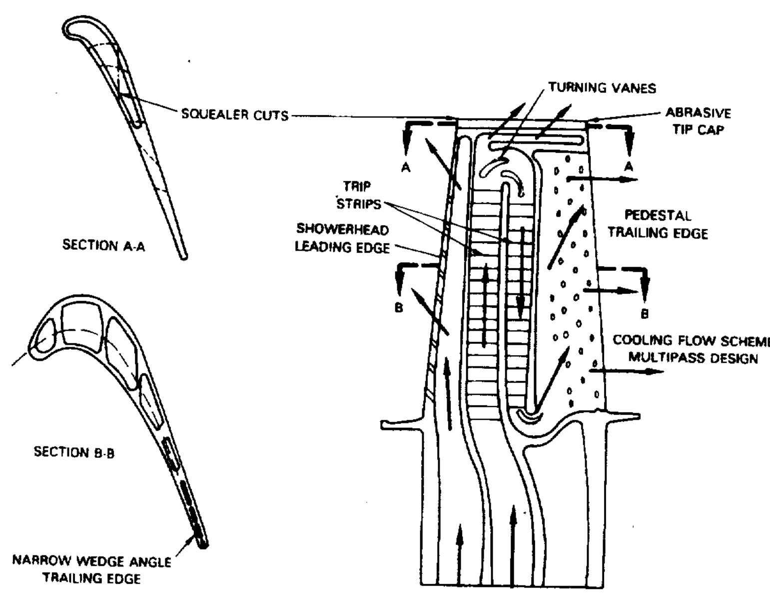


Figure 3-1 High-Pressure Turbine Blade Cooling Scheme

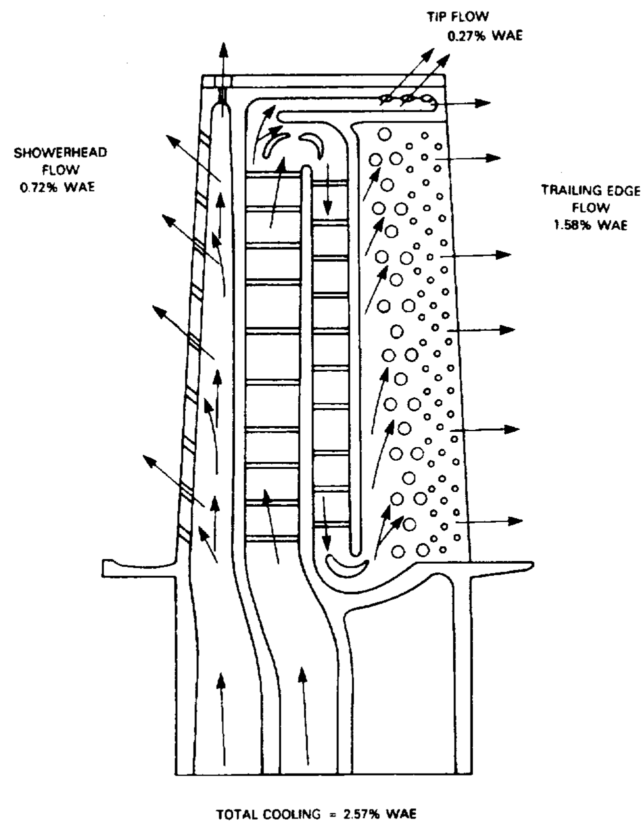


Figure 3-2 Desired Coolant Flow Distribution For High-Pressure Turbine Blade

3.2 TWO-DIMENSIONAL MODELS

Since the root and tip turn flow areas were most critical, initial analysis and design efforts concentrated on two-dimensional models, depicted in Figure 3-3, which would accurately simulate the flow in those regions. To improve the visual assessment of the internal flow distribution, both models were designed five times the actual size of the blade channels. Care was taken to maintain the same flow Reynolds number in the model as would be predicted in the actual blade. Provisions for pressure taps and dye injection ports were incorporated into the model designs at strategic locations in the flow channels.

3.2.1 Tip-Turn Model

The tip-turn model design is illustrated in Figure 3-4. This model includes the outer 25 percent span of the second and third blade cooling passages to simulate the inlet and exit flow conditions at the turns. The channels are trapezoidal in shape and represent the true channel cross-sectional geometry and area variations within the simplifying constraint of flat working materials. Scaled trip strips are incorporated to create the correct degree of channel flow blockage.

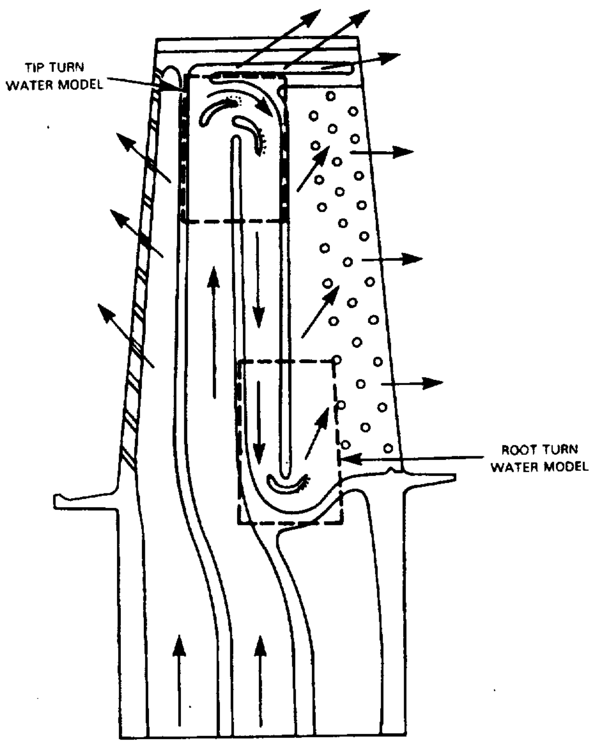


Figure 3-3 Root and Tip Turn Flow Areas Simulated By Two-Dimensional Flow Visualization Models

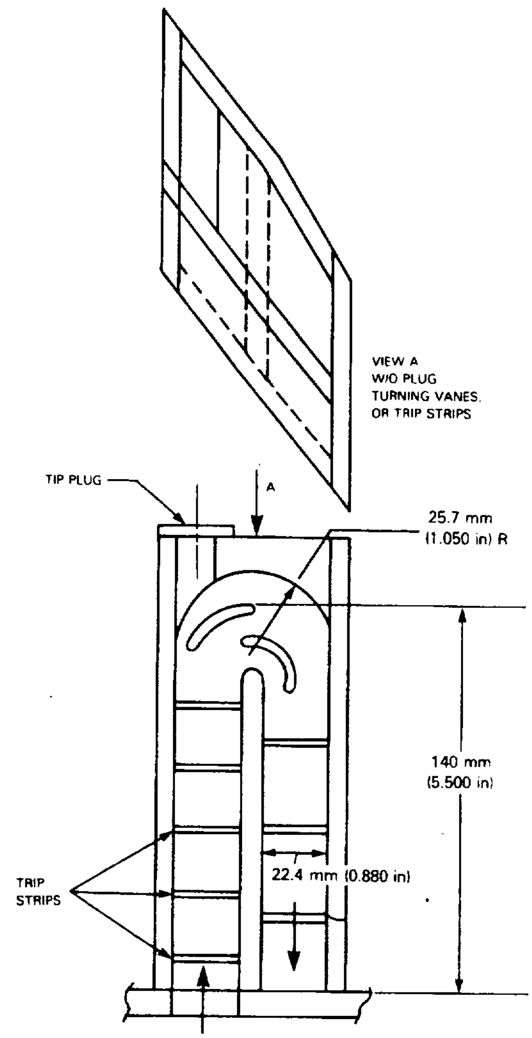


Figure 3-4 Tip-Turn Water Flow Model Design

The model incorporates three removable features to facilitate test evaluations. These include (1) two constant thickness, curved turning vanes, (2) two tapered turning vanes, and (3) a tip plug. The different vane geometries permit evaluation of geometric effects on vane effectiveness and the tip plug permits simulation of cooling flow exiting through the blade tip. Turning vane design details are shown in Figure 3-5 and details of the removable tip plug are shown in Figure 3-6. Provisions for dye injection ports and pressure taps are shown in Figure 3-7.

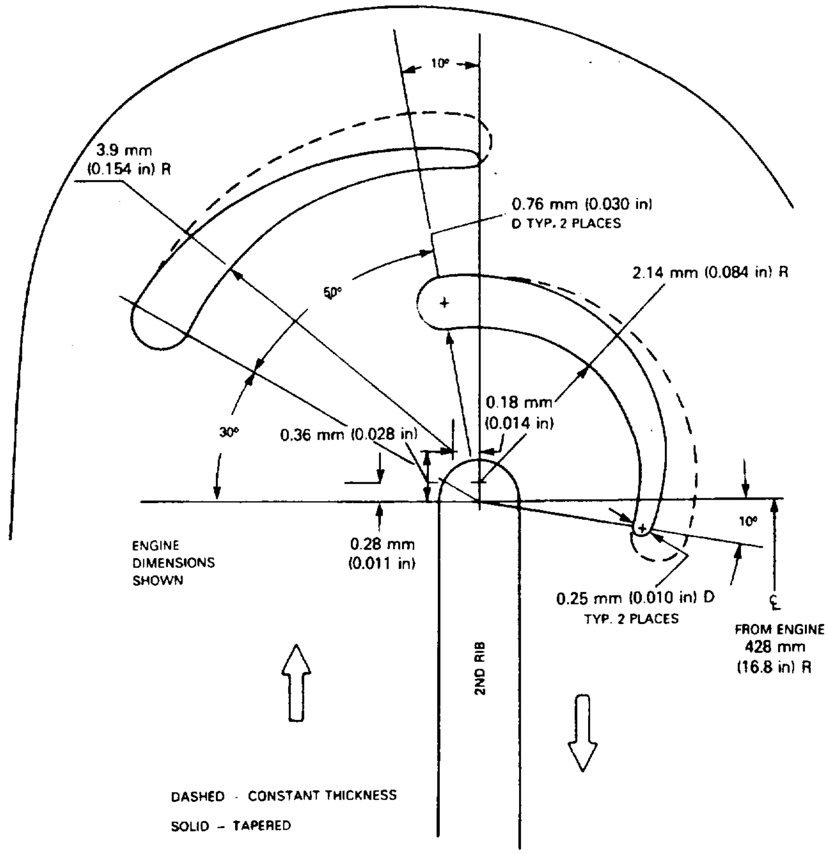


Figure 3-5 Tip Turning Vane Details

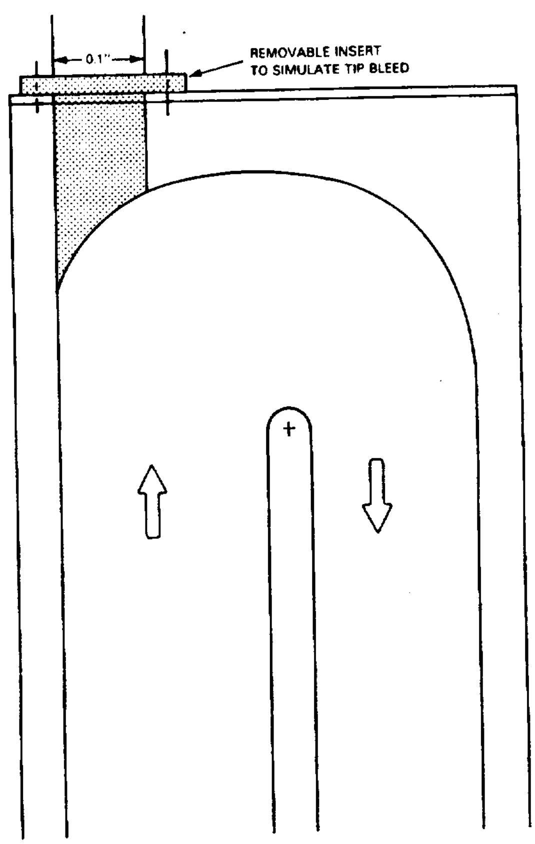


Figure 3-6 Water Flow Tip Turn Plug

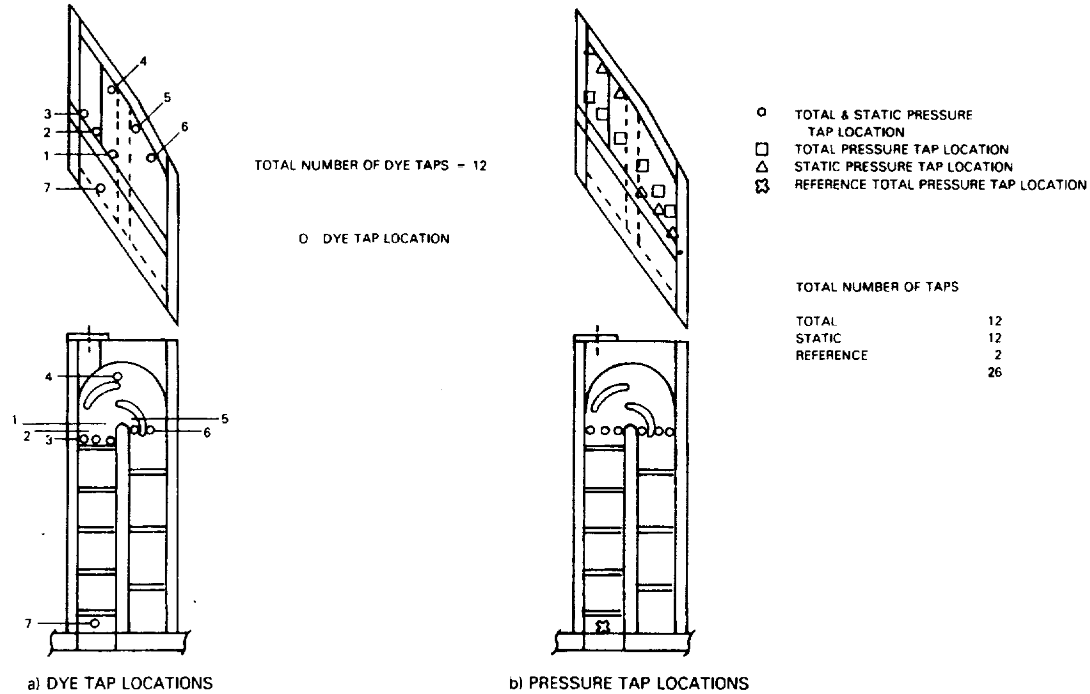


Figure 3-7 Dye Injection Ports and Pressure Tap Locations For Tip-Turn Model

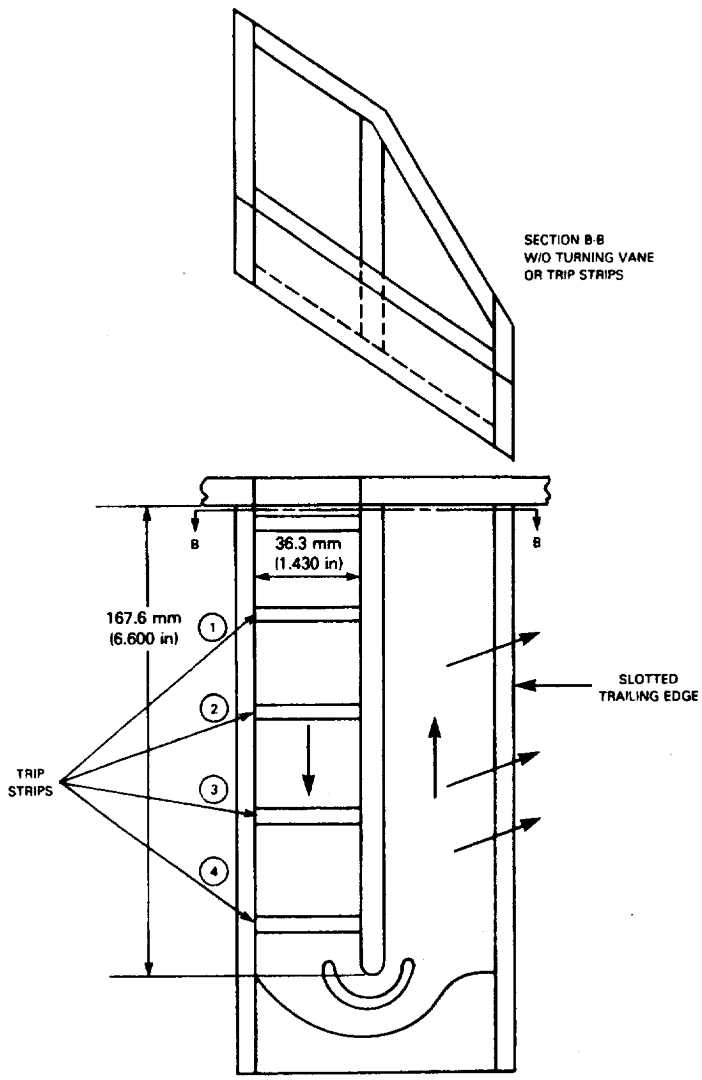
3.2.2 Root-Turn Model

The root-turn model design is illustrated in Figure 3-8 and has internal features similar to those of the tip-turn model. Trip strips are included to create the correct degree of channel flow blockage. The trailing edge wall is slotted to permit simulation of cooling flow bleeding off through the pedestal region. The turning vane is removable so that both constant thickness and tapered turning vane designs can be evaluated. Details of the root-turn geometry are shown in Figure 3-9. Provisions for dye injection ports and pressure taps are shown in Figure 3-10.

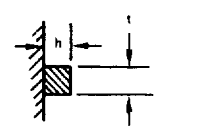
3.3 THREE-DIMENSIONAL MODEL

The analysis and design effort for the three-dimensional model concentrated on duplicating the final design high pressure turbine blade internal cooling geometry in a plastic material suitable for model testing. Salient features of the internal cooling flow geometry associated with this configuration are shown in Figure 3-11.

Details of the three-dimensional flow model are illustrated in Figure 3-12. Like the two-dimensional model, this model was designed at five times the actual blade size to facilitate flow visualization. It incorporates modifications from two-dimensional testing as well as a turning vane configuration that evolved from studies outside the scope of this contract. The model incorporates several internal features to create the required degree of channel flow blockage. These include trip strips, draft angle on the ribs, spanwise area distribution, and an array of trailing edge pedestals.



TRIP STRIP DIMENSIONS



TRIP STRIP	t	h
1	0.2 in (5.08 mm)	0.2 in (5.08 mm)
2	0.2 in (5.08 mm)	0.2 in (5.08 mm)
3	0.3 in (7.62 mm)	0.3 in (7.62 mm)
4	0.3 in (7.62 mm)	0.3 in (7.62 mm)

Figure 3-8 Root-Turn Water Flow Model Design

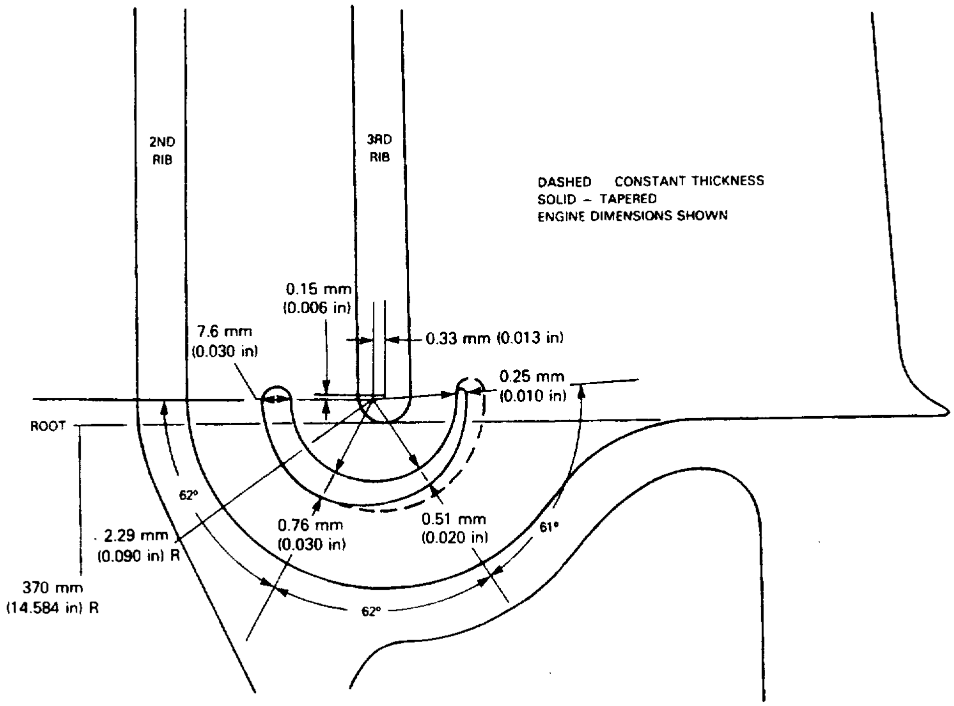


Figure 3-9 Root Turning Vane Details

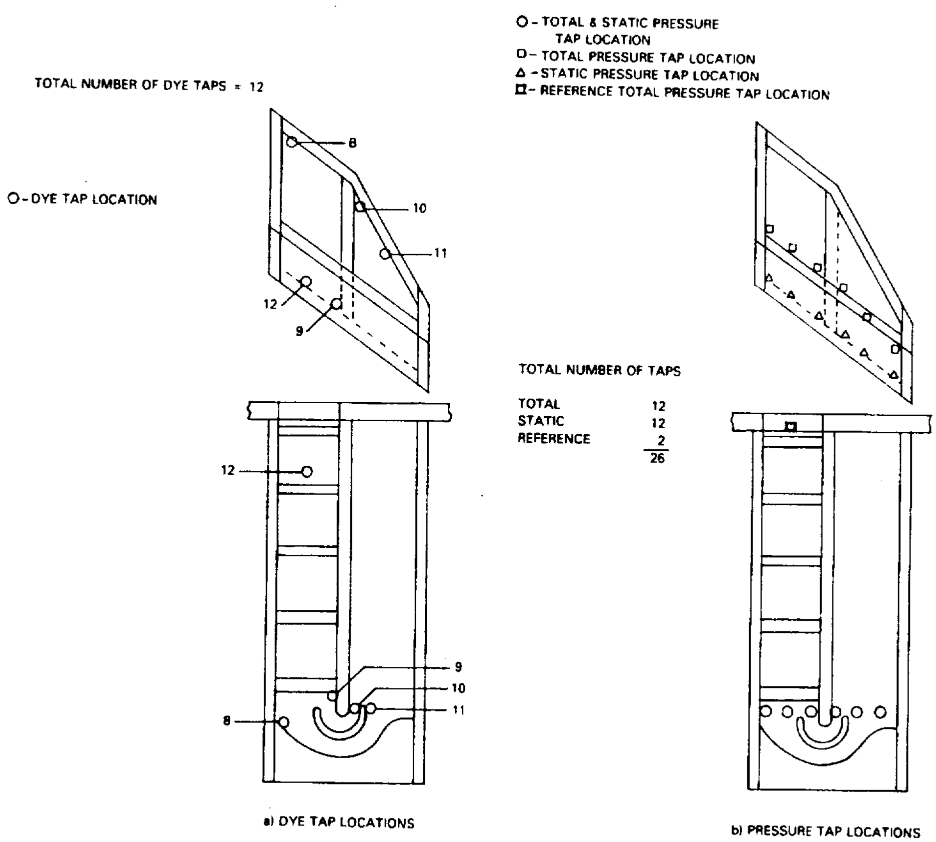


Figure 3-10 Dye Injection Ports and Pressure Tap Locations For Root-Turn Model

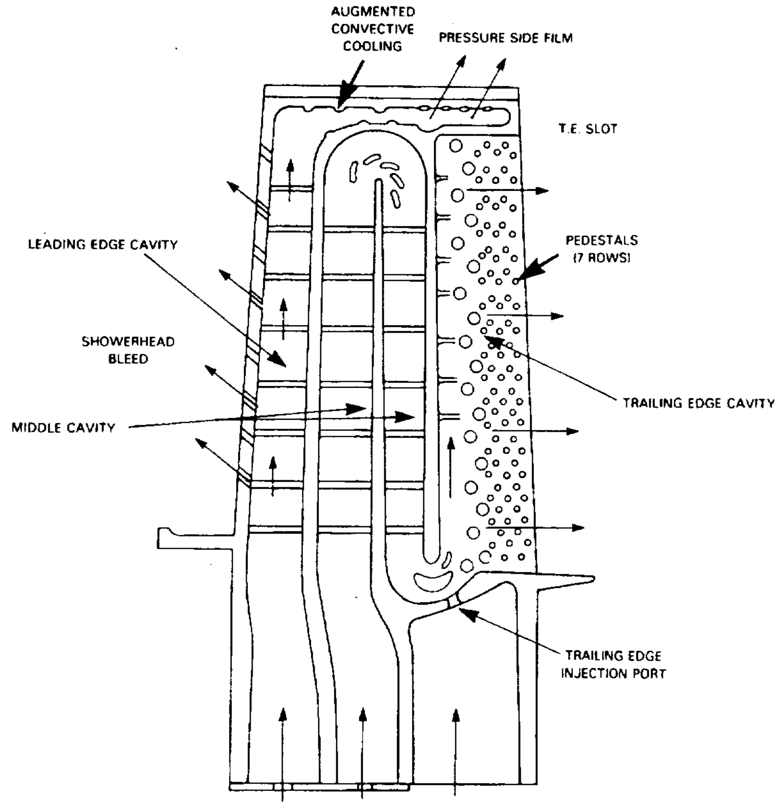


Figure 3-11 Component Blade Cooling Geometry Used To Define Three-Dimensional Water Flow Model

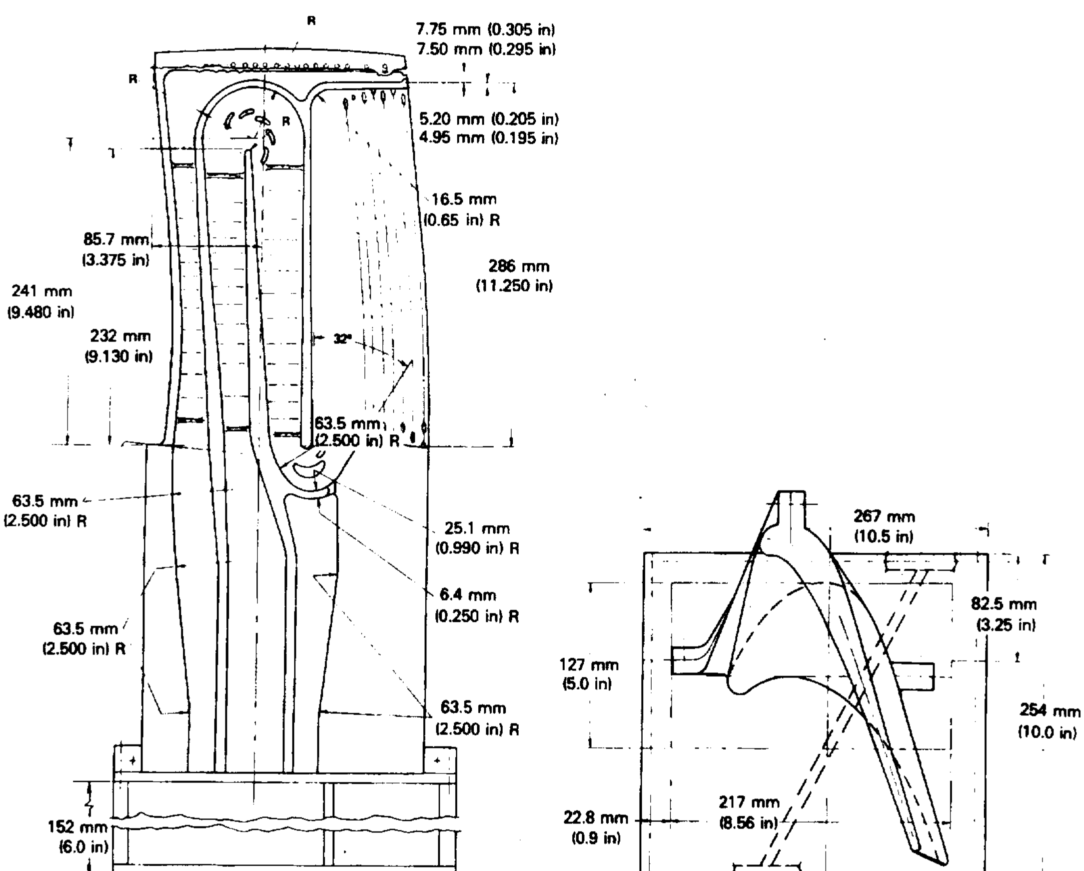


Figure 3-12 Three-Dimensional Flow Model Design Details

Care has been taken to accurately duplicate the flow Reynolds numbers expected in the cooling passages of the actual component blade design. Provisions in the design of the base plenum permit the required flow splits to be metered to the cooling flow passages. Provisions for dye injection ports and pressure taps are shown in Figure 3-13.

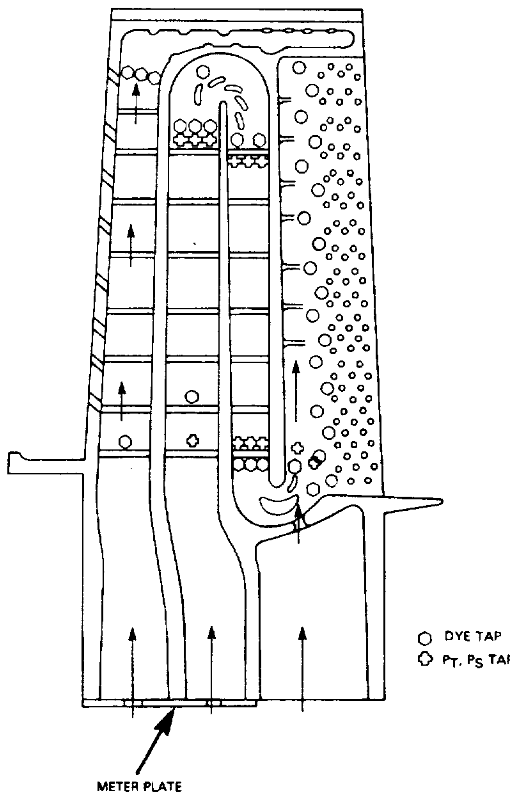


Figure 3-13 Dye Injection Port and Pressure Tap Locations For Three-Dimensional Model

4.0 FABRICATION AND ASSEMBLY

4.1 TWO - DIMENSIONAL MODELS

Two water flow models (Figures 4-1 and 4-2), which simulate the blade internal cooling passages, were constructed using conventional fabrication techniques at five times actual size from plexiglass plate for ease in viewing the internal flow distribution. The models were constructed in sections such that they could be separated and internal modifications incorporated.

4.2 THREE - DIMENSIONAL MODEL

The three-dimensional water flow model is illustrated in Figure 4-3. All internal flow passage details were scaled (5X) to the actual blade tolerences. The internal configuration also included all radius features and draft angles which would be in the actual blade. The outer airfoil shape is constructed of transparent plexiglass which has been cast to shape. The internal shapes are constructed from a "solid" shaped figure which was made utilizing the actual blade detailed design layouts. From this "solid" shape, the ribs, which define flow passages, are machined. The pedestals are constructed from rod stock. The blade model is fabricated in two sections which allows the model to be separated and internal modifications incorporated. In addition, airfoil cooling holes are drilled into the model, simulating actual blade holes.

Figure 4-1 Assembled Two-Dimensional Tip-Turn Water Flow Visualization Model

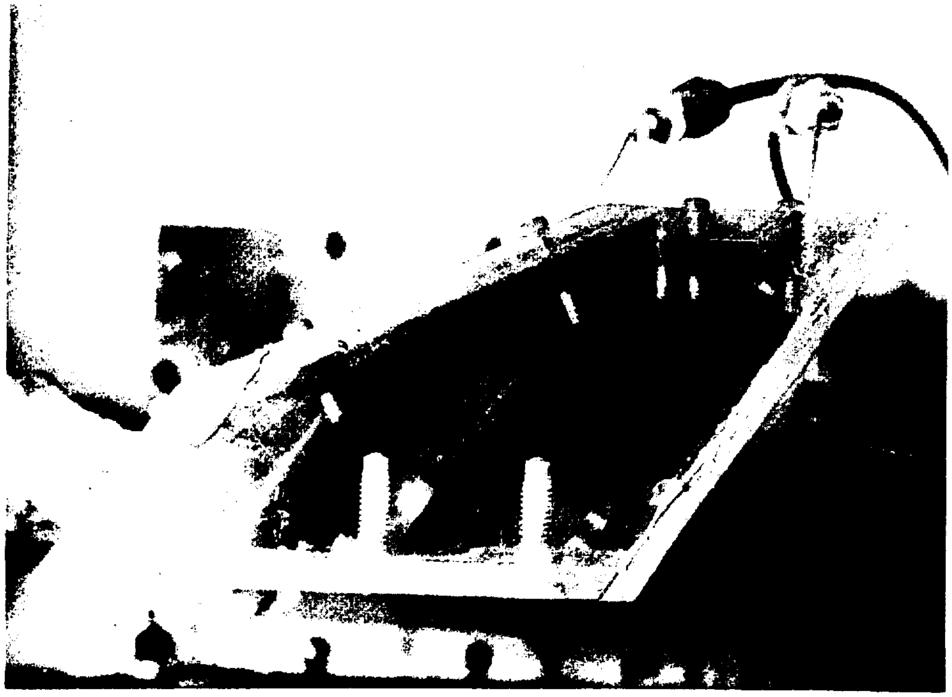


Figure 4-2 Assembled Two-Dimensional Root-Turn Water Flow Visualization Model

Figure 4-3 Energy Efficient Engine - High Pressure Turbine 1st Blade 5X Plastic Flow Visualization Model Pressure Side

5.0 TESTING

5.1 GENERAL DESCRIPTION

Visualization testing is conducted by injecting colored dye at selected locations in the cooling model. Such locations usually are chosen on the basis of previous experience with attention given to turn entrances/exits, potential stagnation or separation locations, and sudden changes in cooling passage geometry.

Flow patterns are then indicated by the presence (or absence) of dye coloration in water and by the relative rates of color movement when the dye injection is started and stopped. Desirable flow fields are characterized by uniform flow velocities and result in uniform mixing and clearing of the dye in water. In contrast, undesirable flows are indicated by uneven or lingering dye patterns characteristic of separating or stagnant flow.

When undesirable flow patterns are observed, the testing is interrupted, a fix is empirically defined and incorporated into the model, and the procedure repeated until acceptable flow is obtained. On new designs such as the Energy Efficient Engine blade, some form of diagnostic testing is required to supplement the qualitative results of visualization and help define fixes. The usual method involves mounting pressure taps at key locations in the model and comparing pressure measurements with assumed design values.

5.2 TEST FACILITY AND INSTRUMENTATION

All tests were conducted at the X-205 stand at Pratt & Whitney Aircraft's Willgoos Laboratory, a facility devoted entirely to the testing of advanced gas turbine engine concepts. The primary components of the facility are: 1) water supply system, 2) water tank to hold model, 3) colored dye injection system, and 4) water to air data system. The existing stand instrumentation fully satisfied all test requirements.

The W.A.D.S. (Water to Air Data System) is a system designed to read small difference of total pressures, in flow rigs, using water as the pressure medium. It is designed so that a back pressure of air is allowed to force the water from the probe and rig reference. until an equilibrium is reached. The principle being that the air pressure required to keep water from entering the probe (or reference) is equal to the water pressure forcing water into the probe.

The system uses a quartz manometer as the readout device. The quartz manometer is connected to read differential pressure, with the high side connected to rig reference which is the highest pressure in the rig and the low side connected through a scanning valve to the probes. Thus, each probe is read as a delta pressure from the rig reference.

The system also reads total pressure, in gage pressure, where the low side of the quartz manometer is opened to atmosphere and the pressure by the air-water equalization method is applied to the high side of the quartz manometer.

In addition to this data acquisition system, the stand has a Polarvision camera recording system. This is a hand held camera which documents the dye flow as it is injected into the water flow model. The film is processed in a cassette and can be played back on a portable screen system. This is useful for post test analysis.

5.3 Test Procedures

5.3.1 Shakedown Testing

Prior to the actual water flow test, the model must undergo a shakedown test to ensure there are no leaks. Since the models are constructed of separate pieces and held together with bolts, external leakage may occur. In addition, internal leaks may occur. These leaks must be repaired as they would contribute to erroneous results. Once the model has been corrected, testing is ready to start.

5.3.2 Two-Dimentional Model Testing

Test Objectives

The flow characteristics of the Energy Efficient Engine high-pressure turbine blade root and tip turns were investigated in this test program. Specific program objectives were to:

(1) measure the pressure drop of the Energy Efficient Engine first stage high-pressure turbine blade root and tip turns,

(2) qualitatively assess the flow characteristics of both turns using flow visualization, and

(3) evaluate the effectiveness of turning vanes in reducing both pressure loss and flow non-uniformities.

Test Conditions

To obtain valid evaluation of blade flow conditions in rig tests, simulation of cooling air Reynolds numbers in the engine was required. Therefore, the channel water flow conditions were selected to match the water flow model Reynolds numbers to engine hot day sea level take-off Reynolds number. The following relationships were used to achieve this match:

$$Re_H = \frac{W_H \; D_H}{A_H \; \mu H} = \frac{W_E \; D_E}{A_E \; \mu E} = Re_E$$

or

$$W_H = W_E \times \left[\frac{A_H}{A_E}\right] \times \left[\frac{D_E}{D_H}\right] \times \left[\frac{\mu H}{\mu E}\right]$$

$$= W_E \times 25 \times (1/5) \times \frac{\mu H}{\mu E}$$

$$= 5 \; W_E \left[\frac{\mu H}{\mu E}\right]$$

where:

R_{e_H} = Reynolds no. in model

R_{e_E} = Reynolds no. in engine

W_H = Water flow rate in model

W_E = Cooling air flow rate in engine

D_H, D_E = Hydraulic diameter

$_H$, $_E$ = Viscosities

A_H, A_E = Channel areas

Testing Sequence

The test procedure for the two-dimensional tip turn cooling model was to conduct the visualization and internal pressure measurement tests on the original version of the model, which included square fillets (no smooth radii) and no turning vanes, and then to proceed with subsequent modifications to the model. The specific test sequence is as follows:

- o test original model
- o add turning vanes
- o add fillet radii
- o substitute airfoil shaped turning vanes (tapered thickness)

Tests were similarly conducted on the root turn model. The succession of tests conducted as part of the original test plan was:

- o original model (no turning vane)
- o add one turning vane

Subsequent analysis of the results of the above tests indicated need for further improvement in the root turn design so the following additional tests were conducted:

- o increased fillet radius in acute corner (three dimensional effect)
- o added cooling air feed directly to trailing edge
- o reduced rib length (increase flow area in the turn).

5.3.3 Three-Dimensional Model Testing

Test Objectives

This test program was conducted to evaluate the final cooling design of the Energy Efficient Engine blade and identify any changes required to eliminate separation or stagnation zones resulting from the final design details not included in the earlier two-dimensional model tests.

The flow characteristics of the newly-defined Energy Efficient Engine blade root and tip turns were investigated. Specific program objectives were to:

(1) qualitatively assess the flow characteristics along both the root and tip turns, and

(2) measure the pressure drop along both turns.

Test Conditions

In order to ensure that the final blade cooling design will function as intended at all high temperature operating conditions, a range of tests covering Reynolds numbers resulting with high, medium and low engine airflows had to be conducted. The following were selected as representative engine operating conditions.

Sea Level Takeoff (standard day)

Maximum Climb (6100mm (20,000 ft)/0.7 Mn/std. day)

Maximum Cruise (11,900mm (39,000 ft)/0.8 Mn/std. day)

Testing Sequence

After completing leak and instrumentation checks, the following test sequence was conducted:

(1) Leading edge/tip cavity and mid chord/trailing edge cavity flow tests covering the range of cooling air flow Reynolds numbers encountered during the noted engine operating conditions.

(2) Identify possible flow separation or recirculation areas in cooling passages and define required pedestal or turning vane modifications to correct deficiencies.

(3) Modify test model.

(4) Repeat step (1) with modified model.

During the course of testing, two additional tests were added as part of diagnostic testing to help identify causes of the small stagnation flow region at the tip of the trailing edge cavity.

These tests included:

(1) measurement of midchord/trailing edge pressure gradients, and

(2) visualization tests with partially blocked trailing edge (to assure that excessive trailing edge discharge flow did not exist and cause the observed stagnant flow area near tip)

5.4 Data Acqusition and Reduction

Basically two types of data were obtained from the tests. The first was by the Polarvision camera which clearly documented injected dye flow patterns as the dye travelled through the water model. The Polarvision system had the advantage of rapid turnaround, as the developed film could be almost instantaneously displayed on the portable screen system. Pressure data were recorded through use of the water-to-air data system which displays the pressure on a digital readout system.

Data reduction was aimed at obtaining: (1) a qualitative assessment of the turn flow characteristics with and without turning vanes, (2) a turn loss coeffi- cient for each scheme, (3) an approximate definition of the turn inlet and exit velocity profiles, and (4) verification of the desired cooling flow distribution.

6.0 RESULTS

Flow visualization was used to inspect airfoil cooling designs for flow separation and stagnation usually not predictable by analysis. Separation is a term describing cooling flow conditions under which the main stream flow does not follow the passage wall, leaving a visible reversed flow or recirculation layer between the main stream and wall. Local flow velocities in such recirculation layers tend to be lower than the main stream velocity. Stagnation implies a very low or complete lack of local flow velocity. Since internal (convective) cooling directly depends on local cooling air velocities, both separation and stagnation cause some local loss of cooling and can result in hot spot metal temperatures during engine operation. The results described in the following sections address these general concerns.

6.1 TWO-DIMENSIONAL MODELS

Testing of the two-dimensional models included pressure loss assessment of the flow in the turns as well as flow visualization to aid in identifying techniques for suppressing the formation of recirculation zones. Unfortunately, measured and calculated pressure losses were often conflicting so that it was difficult to draw consistent conclusions from these data. Consequently, flow visualization was used as the primary basis for determining means for improving the internal flow patterns.

6.1.1 Tip-Turn Model

Flow testing of the original model without turning vanes or smooth corner fillets indicated a moderate degree of recirculation in the acute corner near the tip plug, as shown in Figure 6-1. Also observed was flow separation across the entire backside of the rib and some corner flow recirculation near the flow discharge port at the base of the model.

Addition of two untapered turning vanes, as depicted in Figure 6-2, eliminated the recirculation in the acute corner near the tip plug and eliminated flow separation on the backside of the rib near the airfoil pressure surface wall. However, there was separation in the acute corner near the suction surface wall and in the corner near the flow discharge port.

Corner fillets were added to the model in the acute corners formed at the rib-airfoil surface intersections (shown in Figure 6-3). This reduced the local recirculation in the acute corner at the rib-suction surface intersection, but partial separation still existed along a portion of the rib backside and in the corner near the flow discharge port.

Replacing the untapered turning vanes with tapered turning vanes did not significantly change the degree of flow separation in the acute corner at the rib-suction surface intersection, as shown in Figure 6-4. However, flow appeared to be 'smoothed-out' in the region of the turning vanes. Recirculation persisted at the model base near the flow discharge port.

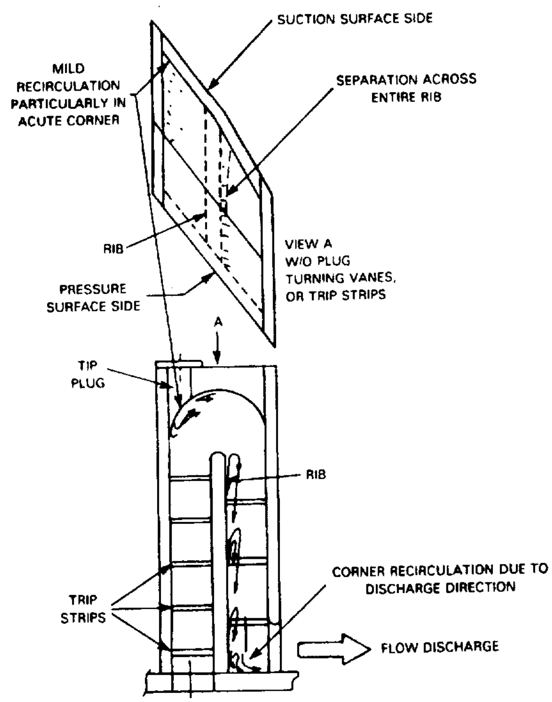


Figure 6-1 Flow Visualization Of Tip Turn Without Turning Vanes

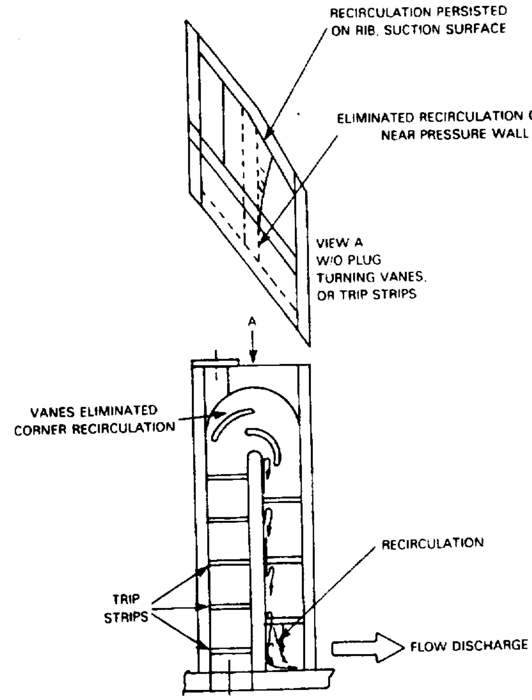


Figure 6-2 Flow Visualization Of Tip Turn With Turning Vanes

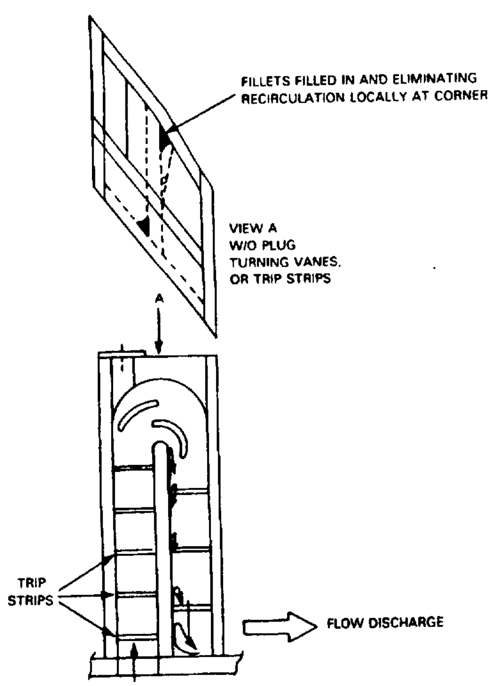


Figure 6-3 Flow Visualization Of Tip Turn With Turning Vanes and Fillets

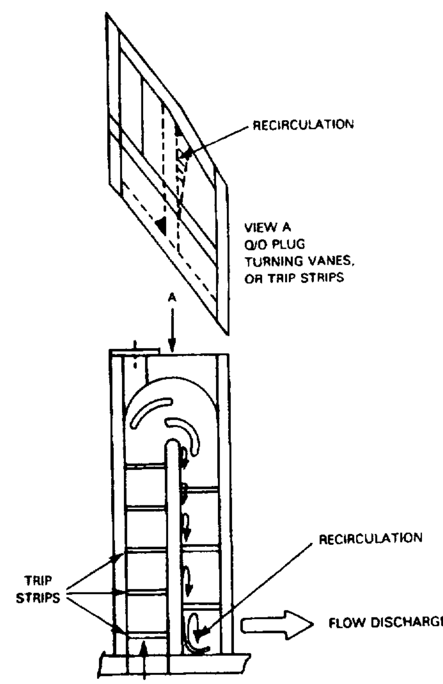


Figure 6-4 Flow Visualization Of Tip Turn With Tapered Turning Vanes and Fillets

The final modification to the model was to change the flow discharge port geometry so that flow exited radially as shown in Figure 6-5. This eliminated the flow recirculation at the base of the model, but a small amount of separation and recirculation existed in the rib-suction surface corner. No further modifications were made since it was felt that the remaining rib flow separation problem could be easily resolved in the design of the three-dimensional model.

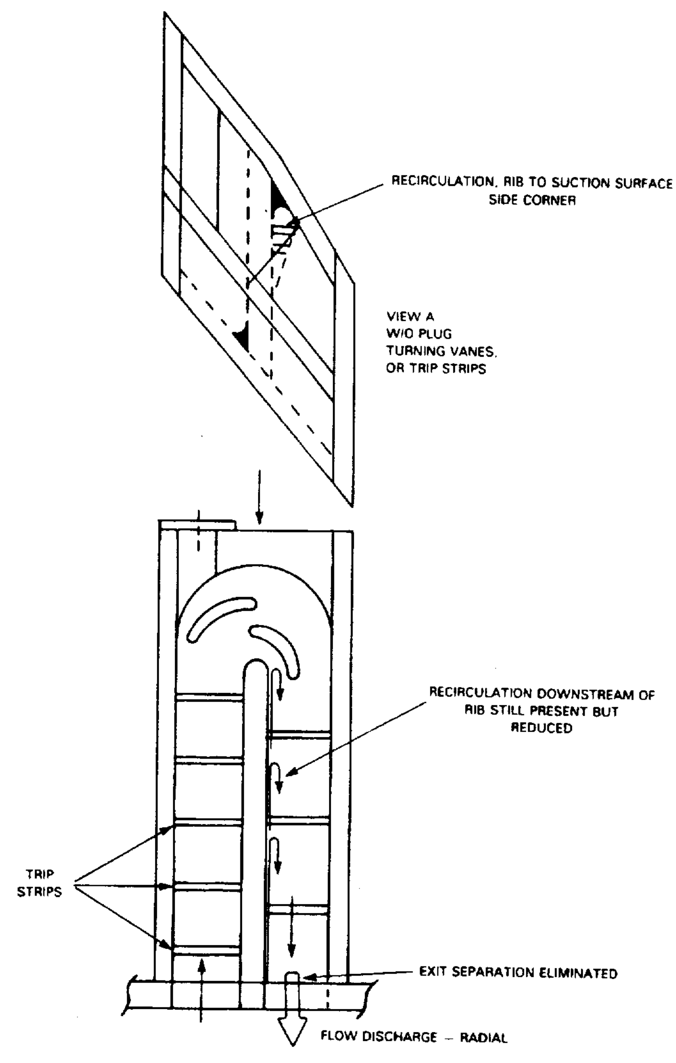


Figure 6-5 Flow Visualization Of Tip Turn With Tapered Turning Vanes, Fillets, and Radial Discharge

6.1.2 Root-Turn Model

Flow testing of the original model without turning vanes or corner fillets indicated flow separation on the back side of the rib in the acute corner formed by the rib and the simulated airfoil suction surface, shown in Figure 6-6. This separation extended along approximately 60 percent of the rib length. In addition, a relatively strong vortex downstream of the last trip-strip just prior to the turn on the front side of the rib appeared to prevent separation of flow in the acute corner formed by the front side of the rib and the simulated airfoil pressure surface.

Addition of a turning vane, as shown in Figure 6-7 did not improve the flow separation problems. As a matter of fact, flow separation along the rib increased to approximately 75 percent of the rib length and a small amount of flow separation was observed on the suction surface of the turning vane downstream of the turn. The trip strip vortex appeared to be less effective.

A fillet was added in the rib-suction surface acute corner, but this had no discernable effect on reducing the flow separation or recirculation in that area or along the rib backside. Consequently, the model was modified as shown in Figure 6-8 so that a portion of the cooling air could be injected directly into the acute corner. At the design flow split of 30 percent flow injected into the trailing edge cavity and 70 percent flow introduced upstream of the root turn, recirculation in the acute corner persisted. Flow split was increased to 50/50 and this did provide some decrease in recirculation. However, a 50/50 flow split is not a viable distribution of flow for the total blade cooling scheme.

The next modifications to the model were designed to increase the flow area at the root turn as well as direct more flow into the acute corner. These modifications are shown in Figure 6-9. They were successful in eliminating acute corner recirculation. However, the 15 degree rib end draft angle (required for good single crystal growth in blade castings) introduced the small region of recirculation on the upstream rib face as illustrated in Figure 6-10. This recirculation zone cleared up as soon as the dye was shut off, indicating that the flow was continuously flowing out of the recirculation zone. It was therefore deemed a minor problem because it can be easily corrected in the component design.

The final modification to this successful model configuration was a restagger of the rib as shown in Figure 6-11. This change reduced the severity of the acute angle corner. The restagger shown best simulates the evolving component design cooling scheme. Flow tests of the model, incorporating this change, indicated that there was no change in the flow field characteristics.

6.2 THREE-DIMENSIONAL MODEL

Testing of the three-dimensional model, as in the two-dimensional models, included pressure loss assessment of flow in the turns as well as flow visualization to aid in identifying techniques for suppressing the formation of recirculation zones. Three flow conditions were simulated in the model; those associated with actual engine operational characteristics at take-off, climb, and cruise power. The model included the final trip strip configuration, draft angles, spanwise core area distribution, and root turn modifications from the two-dimensional test. Tip turns were designed based on results from more current Pratt & Whitney Aircraft model tests.

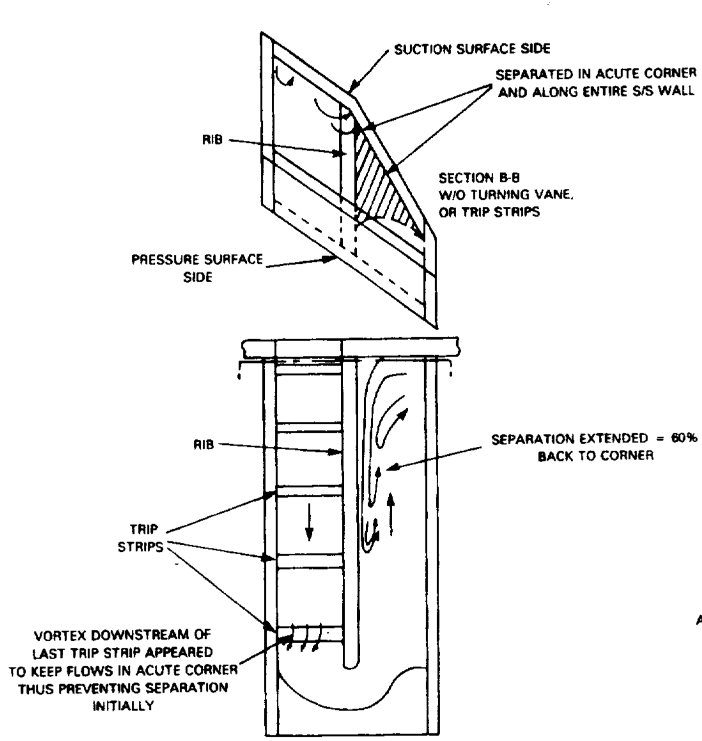


Figure 6-6 Flow Visualization Of Root Turn Without Turning Vane

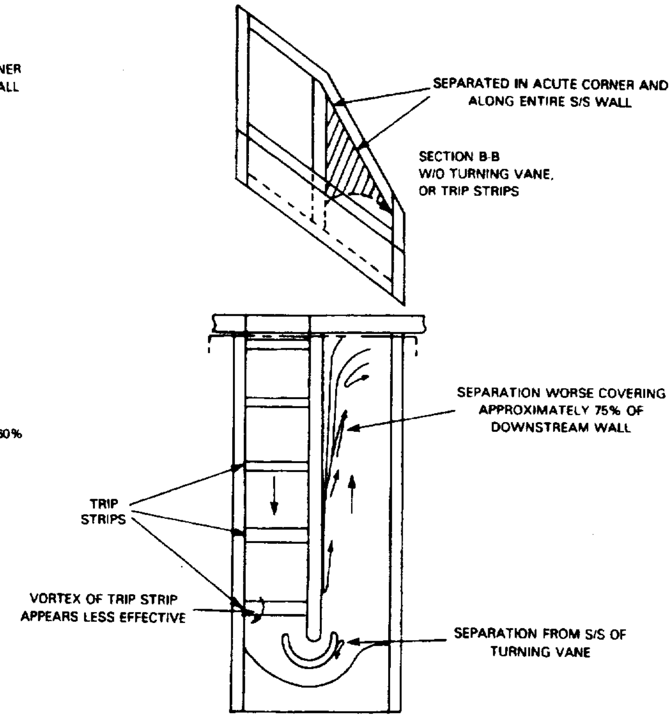


Figure 6-7 Flow Visualization Of Root Turn With Turning Vane

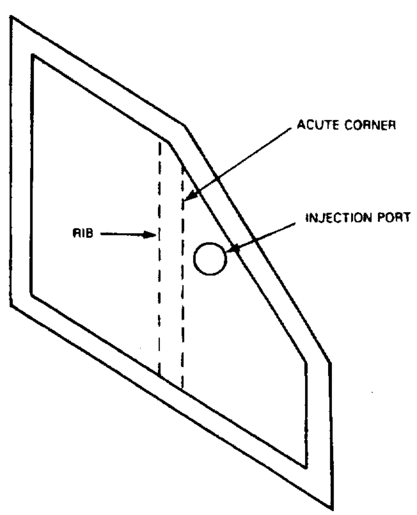


Figure 6-8 Schematic Of Trailing Edge Cooling Flow Injection Port

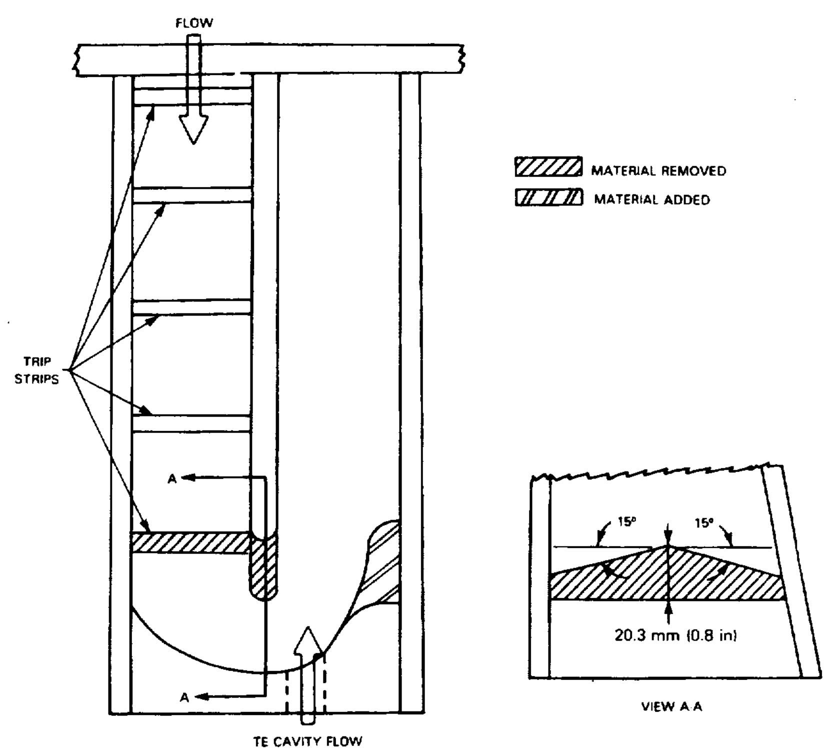


Figure 6-9 Schematic Of Modifications Made To Increase Root Turn Flow Area and Direct More Flow To Acute Corner

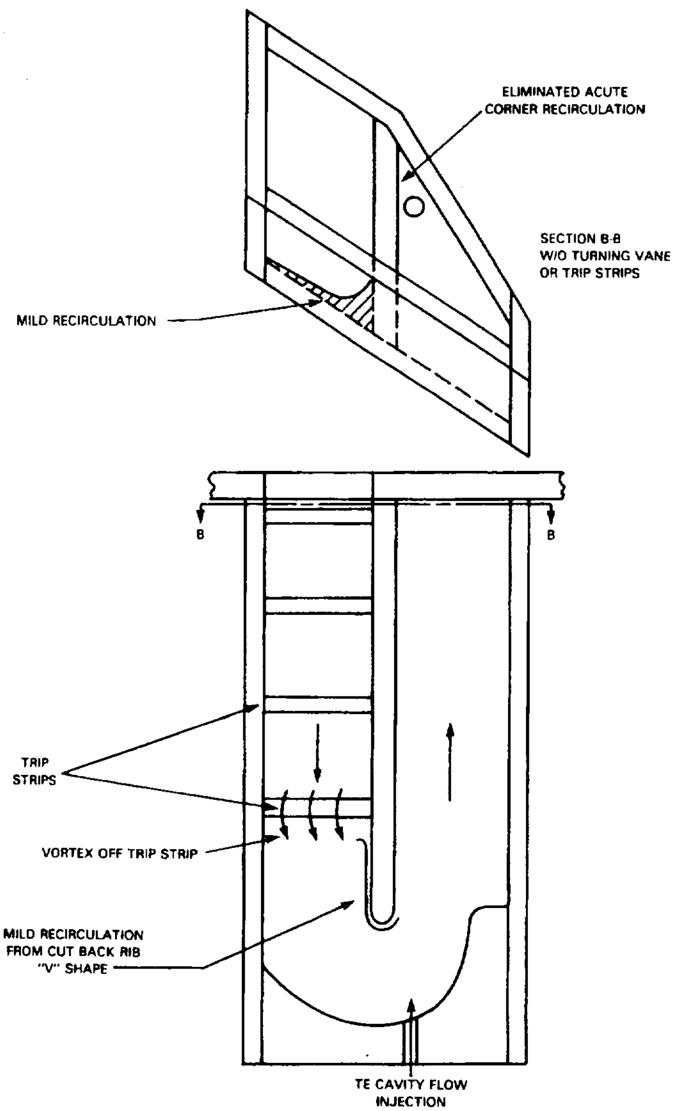


Figure 6-10 Recirculation On Upstream Rib Face Caused By Rib-End Draft Angle

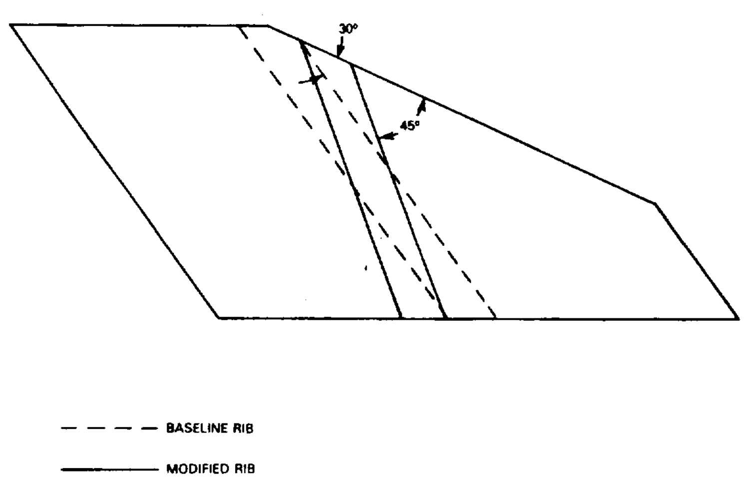


Figure 6-11 Illustration Of Rib Restagger

The initial dye test series yielded the following results. Flow characteristics along the leading edge passage, through the showerhead discharge holes, around the tip passage and out through the tip discharge holes, were all acceptable. Flows up through the multipass cavity around the tip turn, down to the root area and around the root turn were generally acceptable with sporadic recir- culation at the tip vanes. Similar sporadic recirculation was also noted at the root turn and considered acceptable.

The sporadic local recirculation observed at the tip turn (see Figure 6-12) apparently was associated with unstable flow conditions on the convex surface of the first two turning vanes. However, the passage area affected was small and the flow velocities in the recirculation area remained relatively high as indicated by dye clearing rates in the visualization test. Consequently the tip turn was judged to be acceptable.

A larger area of stagnant flow at the tip corner of the trailing edge passage (Figure 6-12) was observed at all three test conditions. The size of the stagnation area did not change with increased flow but the rate at which the dye cleared increased. Subsequently diagnostic testing was conducted to determine causes and provide direction for defining fixes. These tests are discussed in the following paragraphs.

The initial attempt at eliminating the trailing edge recirculation zone was by venting the trailing edge passage to the tip passage. Two venting schemes were evaluated. These are illustrated in Figure 6-13. This was an attempt at a simple fix with minimum tooling change. The success of this approach was limited by the available pressure gradient and the required hole size to achieve sufficient flow velocity. Both analysis and testing subsequently showed that the desired pressure gradient was not achieved and the recirculation zone remained unchanged. The vent holes were then sealed.

Analysis of these results indicated a possible flow starvation problem in this area. Pedestals were added in the trailing edge passage area to provide blockage that would force more flow into the recirculation area. The pedestal arrangements are shown in Figures 6-14 and 6-15. None of these configurations had a noticeable impact on reducing the area of recirculating flow.

The next series of trials consisted of varying degrees of trailing edge blockage in combination with a (2 mm (0.080 in)) diameter flow direction rod angled toward the tip corner through the pedestals. Additional pedestals were incorporated in two of the configurations as shown in Figure 6-16. The first attempt, with 50 percent of the blade trailing edge area blocked, in conjunction with a flow direction rod, showed no effect on the size of the recirculation zone as indicated in Figure 6-16 (A). Next, 30 percent of the trailing edge area was blocked and (6.3 mm (0.25 in)) and (7.9 mm (0.31 in)) pedestals added, as shown in Figure 6-16 (B). This reduced the area of recirculation somewhat. The trailing edge of this configuration was then covered to block 50 percent of the exit area and further reduction in the recirculation area was noted, as shown in Figure 6-16 (C).

Since the partially blocked trailing edge testing did not significantly effect the stagnation area, indications were that sufficient flow could reach the tip corner and that flow starvation in that area was not the cause of the observed flow stagnation.

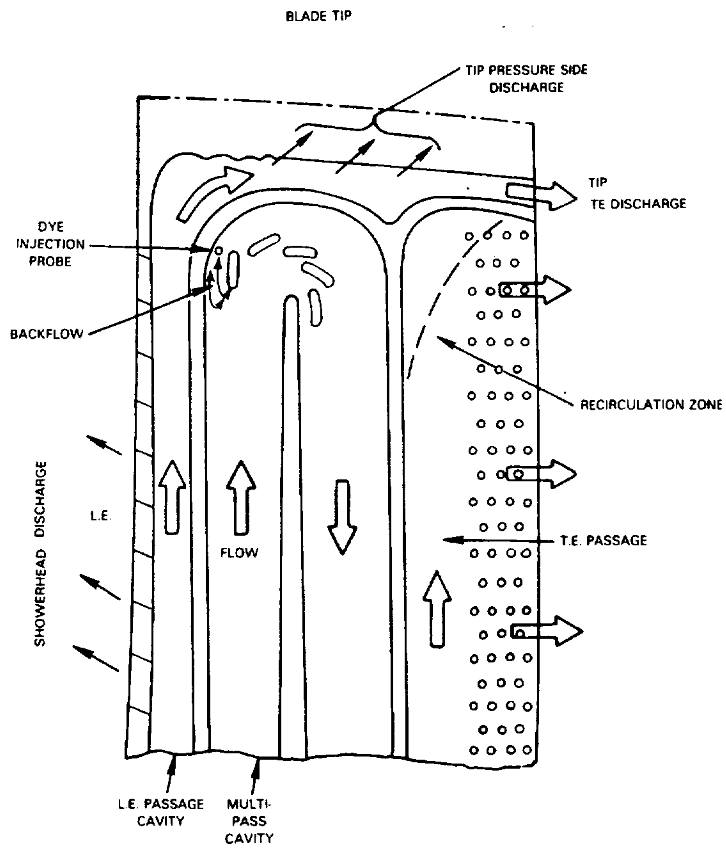


Figure 6-12 Regions Of Recirculation In Initial Three-Dimensional Model Tests

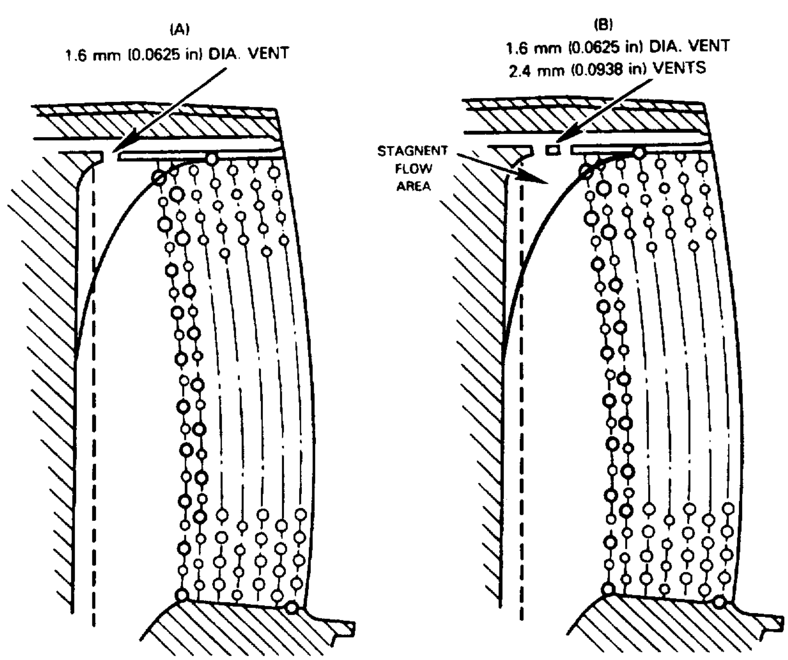


Figure 6-13 Trailing Edge Passage Venting Schemes

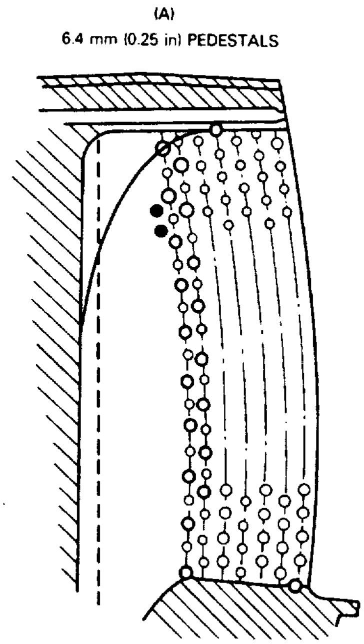


Figure 6-14 Configurations Utilizing 0.25 Inch Pedestals For Flow Control

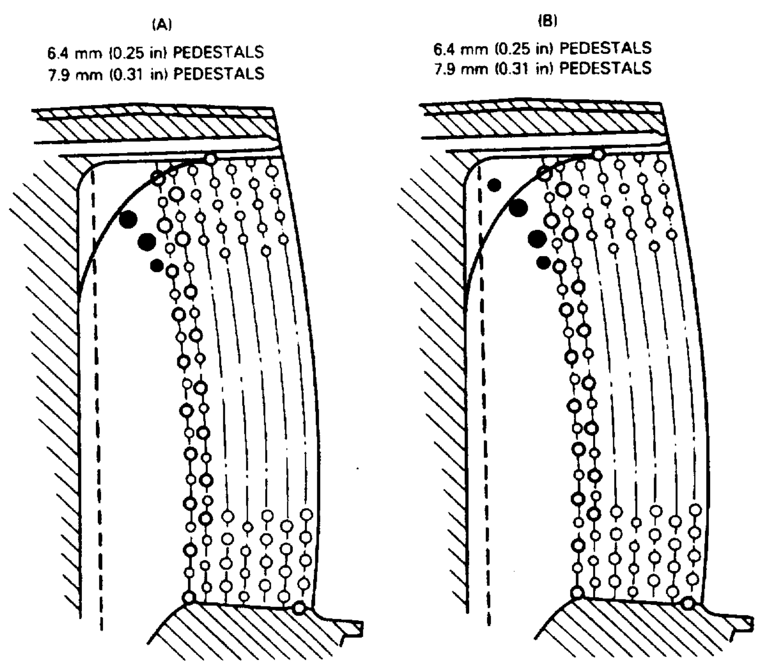


Figure 6-15 Configurations Utilizing a Combination Of 0.25 Inch and 0.31 Inch Pedestals For Flow Control

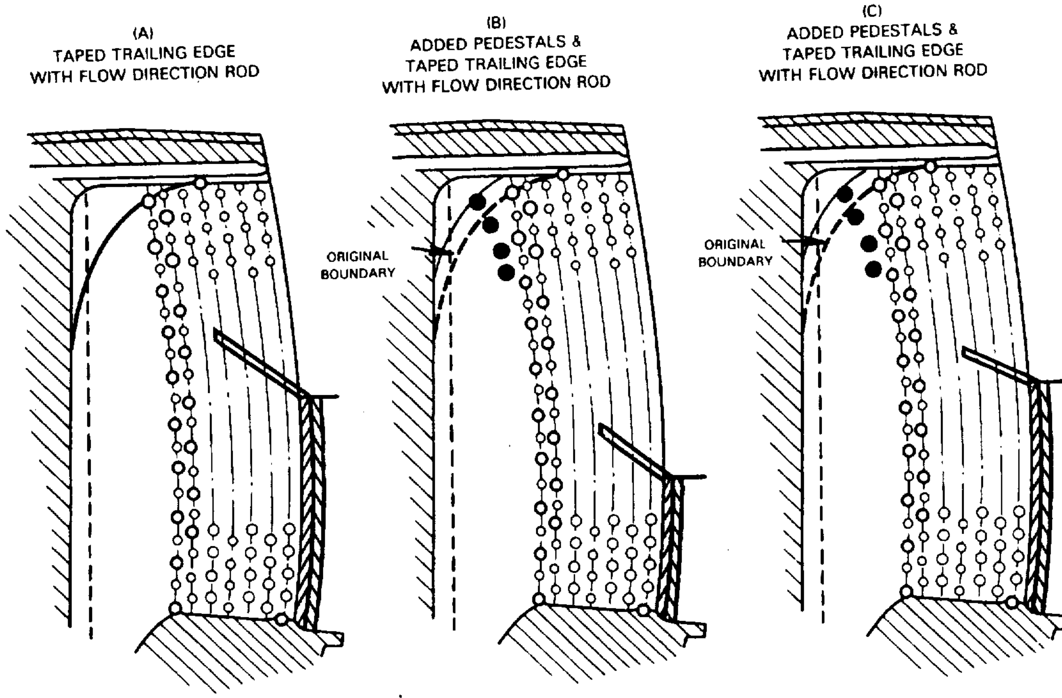


Figure 6-16 Configurations Incorporating Trailing Edge Blockage, a Flow Direction Rod, and Added Pedestals

Subsequent analysis was therefore aimed at the possibility of an undesirable cross-section area distribution in the area of the recirculation. The result of this analysis indicated that the coolant flow passage area normal to the observed flow direction increased sharply near the trailing edge passage tip. This resulted in an estimated equivalent diffusion angle greater than 30 degrees which is sufficient to cause flow separation. With this knowledge in hand, additional pedestals were added in the passage to "smooth-out" the area variation in the divergent region. The initial array of pedestals is shown in Figure 6-17 (A). This solution effectively eliminated the area of recirculation. Further refinements, aimed at minimizing the number of additional pedestals, led to the final pedestal arrangement shown in Figure 6-17 (B). Angling the 5 pedestals to facilitate core withdrawal in the blade fabrication process had no affect on final flow conditions.

As part of the diagnostic testing, pressures were measured at locations considered susceptible to higher than expected losses. These included the tip turn, root turn, and trailing edge channel. The dynamic pressure head at the inlet to the turns, and the turning losses themselves, were calculated from total pressure data. Calculated losses are compared to design assumptions in Table 6-I. Measured turning losses are less than design losses. The root turn loss includes the effect of the trailing edge feed, which tends to reduce the apparent turn loss. An estimate of the loss with this loss subtracted out is also shown in the table. Pressure measurements downstream of the turns indicated some nonunifority in the velocity profiles, especially at the root turn. However, these were considered acceptable.

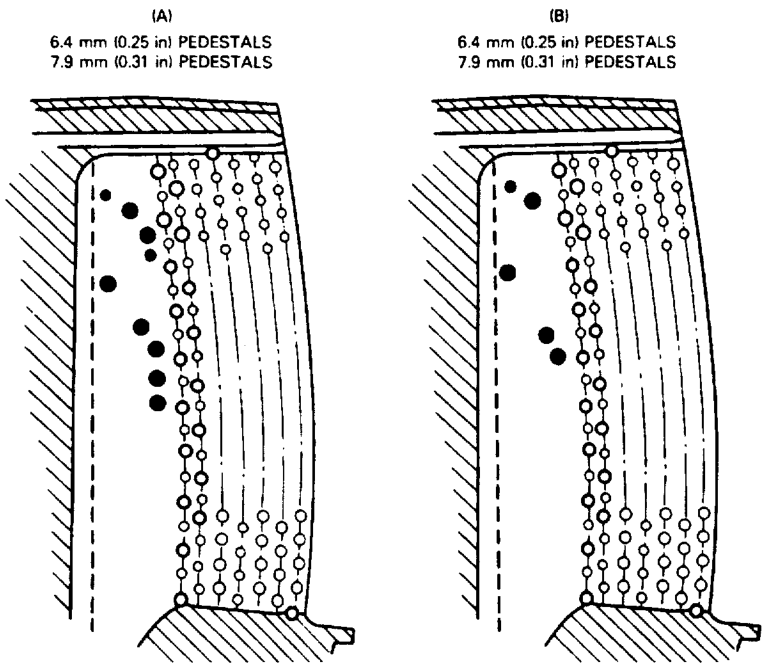


Figure 6-17 Pedestal Arrangements Which Eliminated Flow Recirculation in the Trailing Edge Tip Passage

TABLE 6-I

TURNING LOSSES

	$\frac{\Delta PT}{Q}$ Design	$\frac{\Delta PT}{Q}$ Test
O. D. TURN	2.0	1.75
I. D. TURN	3.5	1.77 w/Rear Feed 2.5 w/o

ΔP_T = Total pressure loss across the turn
Q = dynamic pressure head at entrance to the turn

7.0 CONCLUSIONS

Two-dimensional and three-dimensional water flow model testing confirmed the effectiveness of the high pressure turbine blade cooling geometry, following minor modifications. Conclusions resulting from two-dimensional model tests are as follows:

- o In the tip-turn model, incorporation of constant thickness turning vanes eliminated recirculation in the acute corner of the flow passage near the tip plug and reduced flow separation along the back side of the rib. Addition of fillets in the acute corners formed at the rib-endwall intersections further reduced flow separation along the back side of the rib and significantly reduced flow recirculation in the acute corners. Changing the flow discharge port to exit radially instead of axially eliminated the remaining flow separation along the rib backside. Adding taper to the turning vanes had no visible effect on the observed flow patterns although there was some "smoothing-out" of the flow in the region of the turning vanes.

- o In the root-turn model, flow separation on the rib back side and recirculation in the acute corner formed by the rib and simulated airfoil suction surface were eliminated by increasing the passage flow area at the root turn and by injecting flow directly into the trailing edge cavity near the acute corner. Restaggering the rib to reduce the severity of the acute angle to facilitate blade casting had no adverse affect on flow field characteristics.

Conclusions resulting from the three-dimension model tests are:

- o Flow conditions in the leading edge and tip passages, and at the tip turn and root turn passages were all acceptable.

- o A large zone of recirculating flow at the tip corner of the trailing edge passage was found to be caused by a sharp divergence in the passage flow area near that region. The recirculation was corrected by the addition of 5 pedestals in the affected passage area. These provided the required blockage to eliminate the divergence and associated flow separation.

- o Measured turning losses were lower than predictons but the same order of magnitude.